International Journal of Mechanical Engineering and Technology (IJMET)
Volume 8, Issue 2, February 2017, pp. 210–219 Article ID: IJMET_08_02_026
Available online at http://www.iaeme.com/IJMET/issues.asp?JType=IJMET&VType=8&IType=2
ISSN Print: 0976-6340 and ISSN Online: 0976-6359

COMPUTATIONAL FLUID DYNAMIC ANALYSIS OF AIRFOIL NACA0015

Prof. V.B. Swami
Department of Mechanical Engineering,
Walchand College of Engineering, Sangli, Maharashtra, India

Shubham Dhawale, Onkar More, Rohidas Gude, Suraj Rathod, Nikhil Kudave
UG Students, Department of Mechanical Engineering,
Walchand College of Engineering, Sangli, Maharashtra, India

ABSTRACT

In this chapter we choose standard airfoil NACA 0015. Which is symmetrical airfoil with a 15% thickness to chord ratio was analyzed on ANSYS FLUENT to determine the coefficient of lift, coefficient of drag and graph of coefficient of lift vs. coefficient of drag. The 2-dimensional cross sectional view was considered. The wind velocity was taken as 17m/s which are corresponding to 232,940 Reynolds number. The airfoil, with an 8 in chord, was analyzed at 0, 5, 10 and 15 degrees. Parameters viz. Coefficient of lift (Cl), Coefficient of drag (Cd) and Cl/Cd are calculated and are plotted against different angle of attack.

Key words: Angle of attack, Coefficient of lift, Coefficient of drag, CFD analysis.

Cite this Article: Prof. V.B. Swami, Shubham Dhawale, Onkar More, Rohidas Gude, Suraj Rathod and Nikhil Kudave. Computational Fluid Dynamic Analysis of Airfoil NACA0015. *International Journal of Mechanical Engineering and Technology*, 8(2), 2017, pp. 210–219.
http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=8&IType=2

1. INTRODUCTION

NACA AIRFOIL is developed by the NACA (National Advisory Committee for Aeronautics). The shape of the AIRFOIL is described by using a series of digits following the word "NACA”. The first digit involves the maximum camber as percent of the chord. The second digits involve location of that maximum camber measured from leading edge in percent of the chord and last two digits involves maximum thickness o the AIRFOIL in percent of chord length.

The 2-dimensional cross sectional view is considered. The wind velocity is taken as 17m/s which is corresponding to 2,32,940 Reynolds number [4] . The angle of attack is varied.

Parameters Studied - Coefficient of lift (Cl), Coefficient of drag (Cd) are calculated and are plotted against different angle ofattack.

Software - “ANSYS FLUENT” version 13.1.[1]

NACA airfoils are developed by National Advisory Committee for Aeronautics (NACA).

Angle of attack

If you stretch your arm out through the window of car which is moving with some speed, it feels your arm pushed backward. When you hold your arm straight with your hand parallel to the road, and change the angle slightly, suddenly feel that it is drown upwards. Hand and arm work like the wing of an airplane and with the right angle of attack The Computational Fluid Dynamics (CFD) software used is you can feel a strong lift force.[3]

Chord length The distance between leading edge and tailing edge.

Nomenclature of an Airfoil

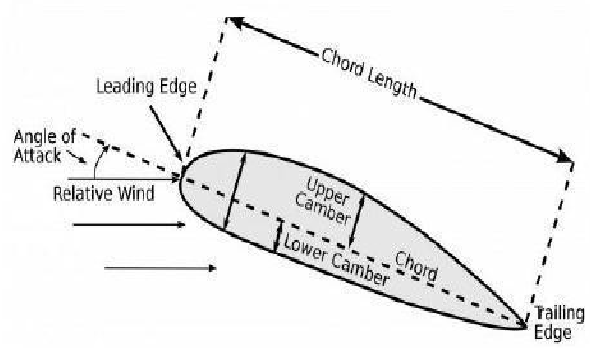

Figure 1 Basic Nomenclature of an airfoil [2]

2. METHODOLOGY

- Pre-analysis & start-up
- Geometry
- Mesh
- Physics setup
- Numerical solution and results

2.1. Pre-Analysis & Start-Up

One of the important things one should think about before one set up the simulation is planning the boundary conditions of the given set up. C-Mesh is One of the popular meshes for simulating an airfoil in a stream. and that is what we will be using. At the inlet of the system, we have defined the velocity as entering at a 0-degree angle of attack and at a total magnitude of 1. We defined the gauge pressure at the inlet is 0. As for the outlet, the only thing we have assumed is that the gauge pressure is 0. As for the airfoil itself, we have treated it like a wall.

2.2. Airfoil Geometry

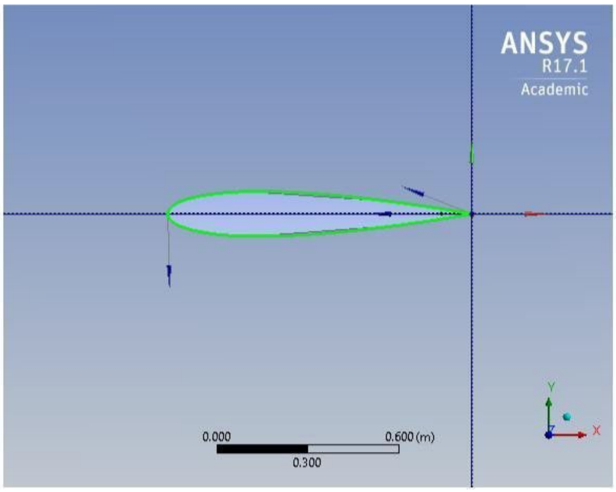

Figure 2 Flow Domain Generation

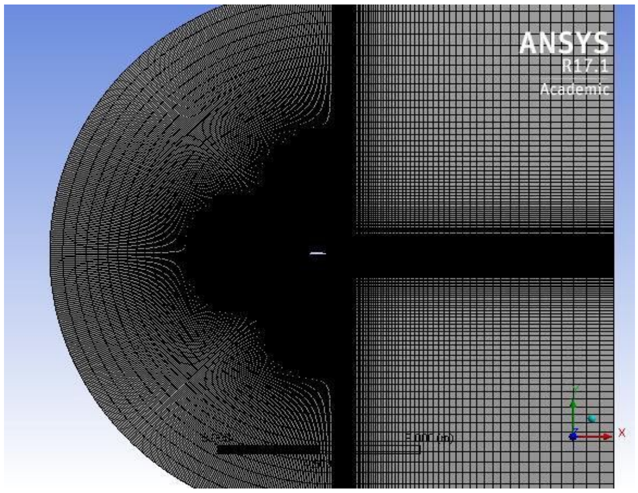

Figure 3 Fine meshed flow domain [4]

2.3. Meshing of Flow Domain

First, we have applied a mapped face meshing control to the geometry. Next, we applied edge sizing [2] controls to all of the edges of the mesh. In order to analyze fluid flow, flow domains is split into smaller sub domains. The governing equations are applied and solved inside each of these sub domains. The meshing around the aerofoil is shown in below figure in which meshing accuracy is increases as we go towards the aerofoil.

Object Name	*Surface Body*
State	Meshed
Graphics Properties	
Visible	Yes
Transparency	1
Definition	
Suppressed	No
Coordinate System	Default Coordinate System
Thickness	1. m
Thickness Mode	Refresh on Update
Behavior	None
Reference Frame	Lagrangian
Material	
Fluid/Solid	Defined By Geometry (Solid)
Bounding Box	
Length X	25. m
Length Y	25. m
Properties	
Volume	557.83 m³
Centroid X	2.1769 m
Centroid Y	-1.6445e-008 m
Centroid Z	0. m
Surface Area(approx.)	557.83 m²
Statistics	
Nodes	60600
Elements	60000
Mesh Metric	None

3. RESULTS AND DISCUSSIONS

In this study, numerical analyses were performed. The analysis was performed at 17 m/s wind velocity. Coefficient of Lift and drag of NACA 0015 airfoil at different angle of attack between 0° and 15° were calculated. The lift and drag coefficients are obtained as numerical with FLUENT programs for the same conditions. In numerical analysis C mesh was used. The top bottom and left boundaries are placed at a distance of 10 chords from airfoil. Whereas the right boundary was placed at 20 chords. A mesh independent study was performed to verify that the solution would not change subsequent additional refinements and 1,35,000 grids number suitable for our model. Airfoils have various shape and sizes. The non– dimensional coefficients for two dimensions' solution is given as below: [1]

Lift coefficient [6]:

$Cl = 2L/\varrho v^2 c$

Drag coefficient [6]:

$Cd = 2D/\varrho a v^2 c$

Where, *L* and *D* are lift, and drag force, *Cl* and *Cd* are lift and drag coefficient of airfoil respectively, *c* is airfoil cord length, *V* is velocity of wind, *ρ* is density of air.

3.1. Results

The lift coefficient varies with angle of attack. Increasing angle of attack is associated with increasing lift coefficient. After maximum lift coefficient, lift coefficient decreases as the angle of attack. A symmetrical wing has zero lift at 0 degrees angle of attack.

Identically the value of drag coefficient is zero at the zero angle of attack and it increase slowly till the stall condition and at the time of stall as well as after stall it increase readily.

The critical or stalling angle of attack is typically around 15° for many airfoils.

Coefficient of drag and lift with iterations at different angle of attack

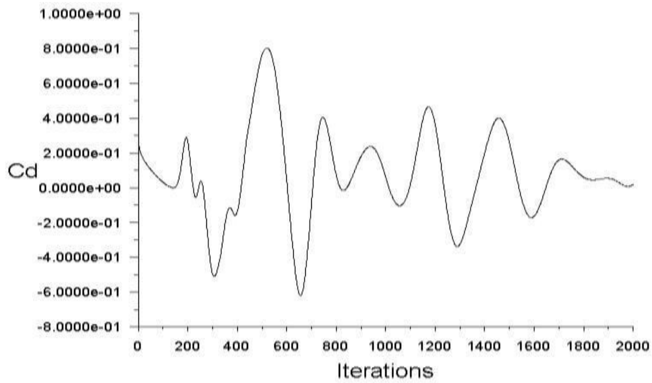

Figure 4 Plot between coefficient of drag and iterations at an angle of attack 0 degree

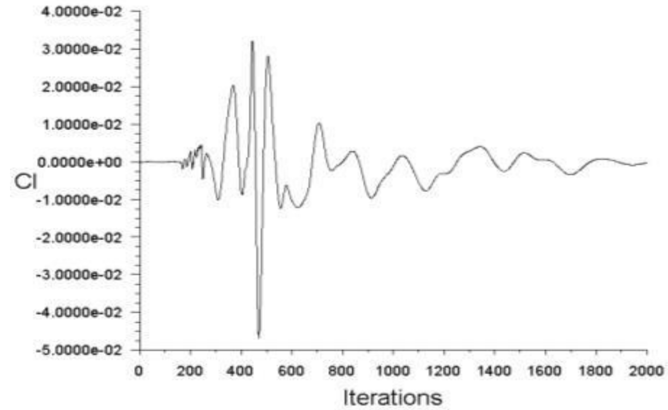

Figure 5 Plot between coefficient of lift and iterations at an angle of attack 0 degree

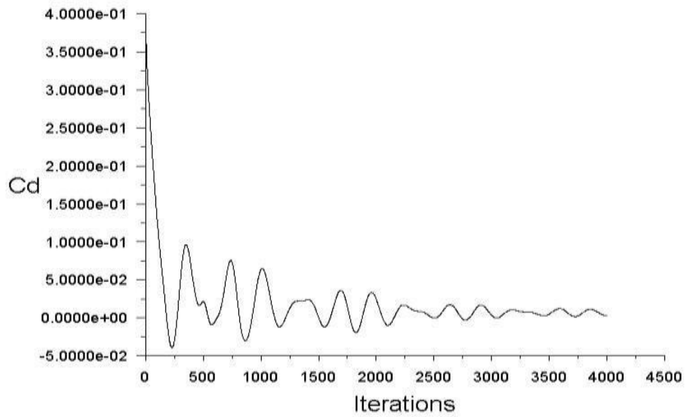

Figure 6 Plot between coefficient of lift and iterations at an angle of attack 5 degree

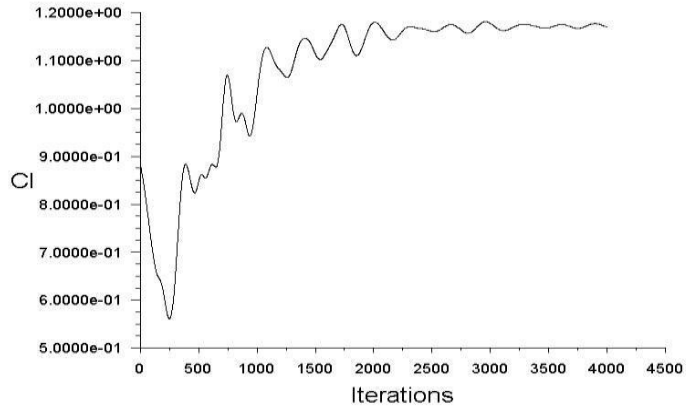

Figure 7 Plot between coefficient of lift and iterations at an angle of attack 10 degree

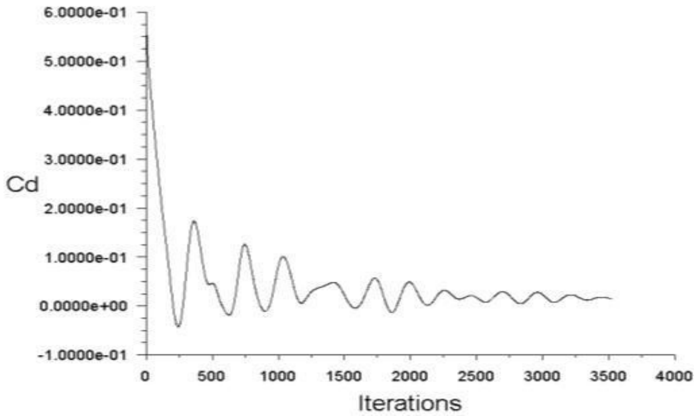

Figure 8 Plot between coefficient of drag and iterations at an angle of attack 15 degree

3.2. Velocity Counters

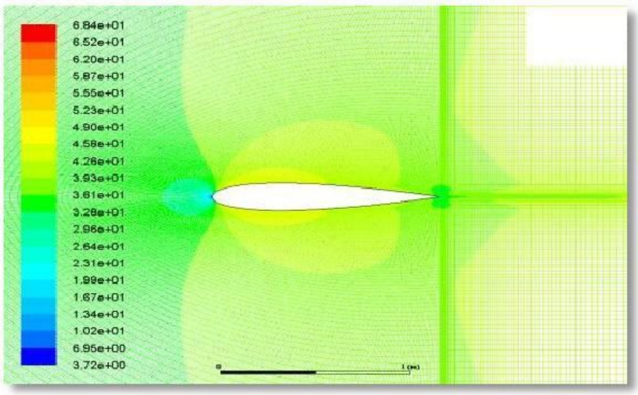

Figure 9 Velocity counter at an angle of attack 0 degree

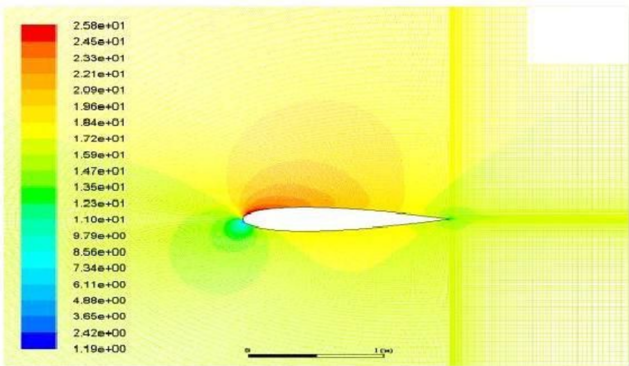

Figure 10 Velocity counter at an angle of attack 5 degree

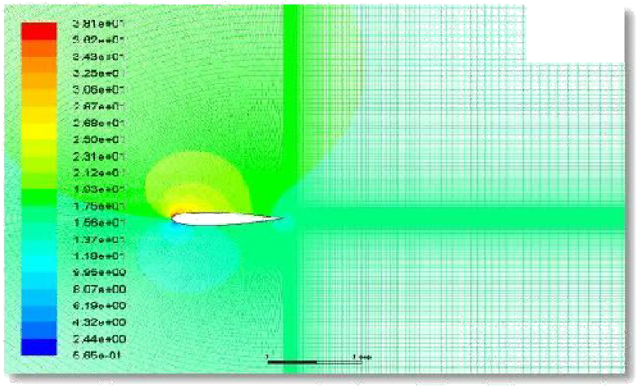

Figure 11 Velocity counter at an angle of attack 10 degree

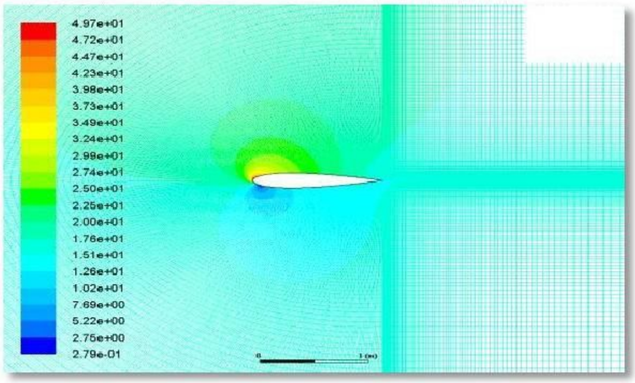

Figure 12 Velocity counter at an angle of attack 15 degree

3.3. Pressure Counter

At zero lift the pressure distributions over the upper and lower surfaces is identical. A contour of static pressure shows that static pressure increases at the lower surface of the aerofoil with increasing angle of attack.

At no incidence, In case of symmetrical airfoil, the distribution of velocity and thus the pressures along both surfaces would have been exactly the same, canceling each other & total lift force is zero.

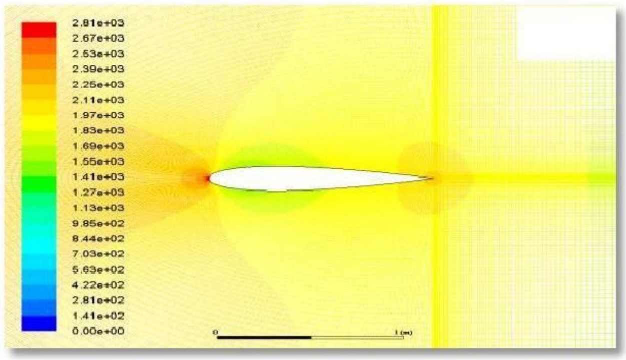

Figure 13 Pressure counter at an angle of attack 0 degree

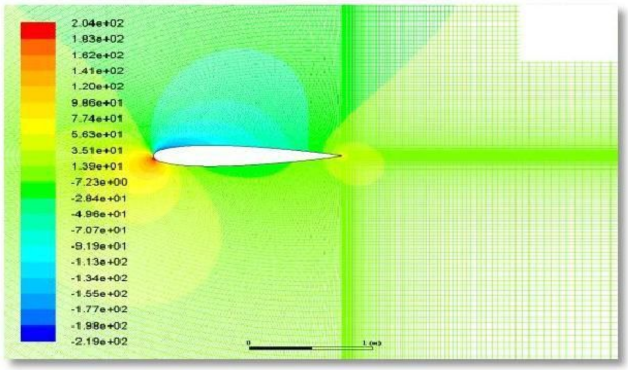

Figure14 Pressure counter at an angle of attack 5 degree

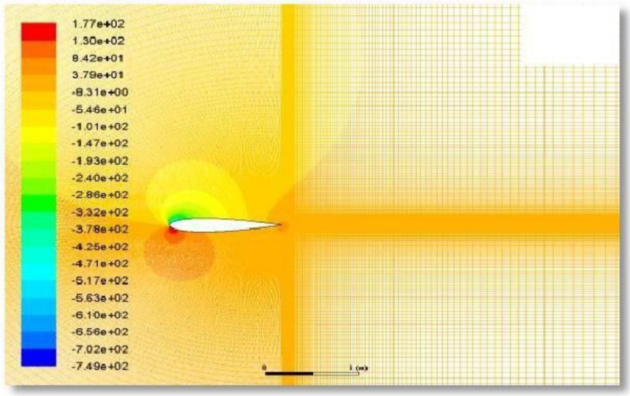

Figure 15 Pressure counter at an angle of attack 10 degree

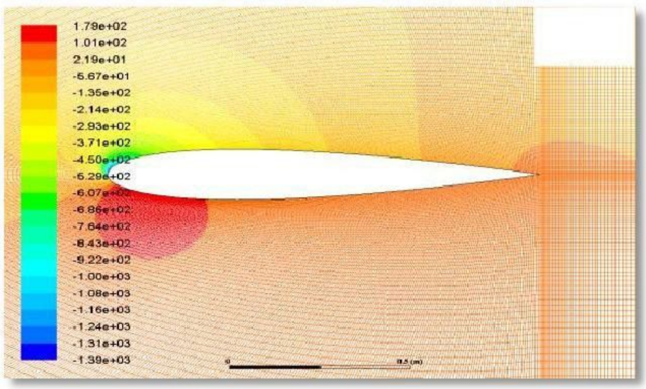

Figure 16 Pressure counter at an angle of attack 15 degree

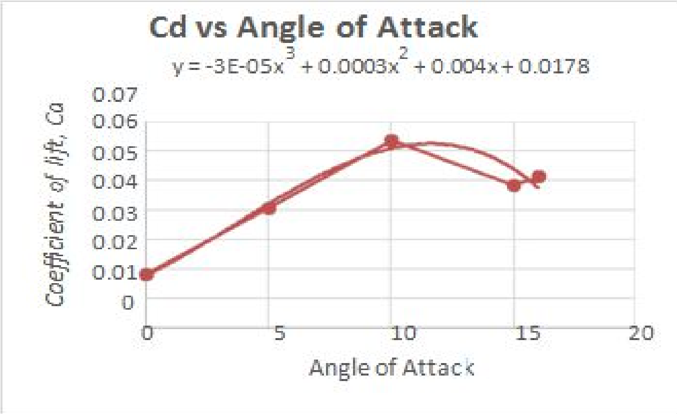

Figure 17 Cd vs angle of attack

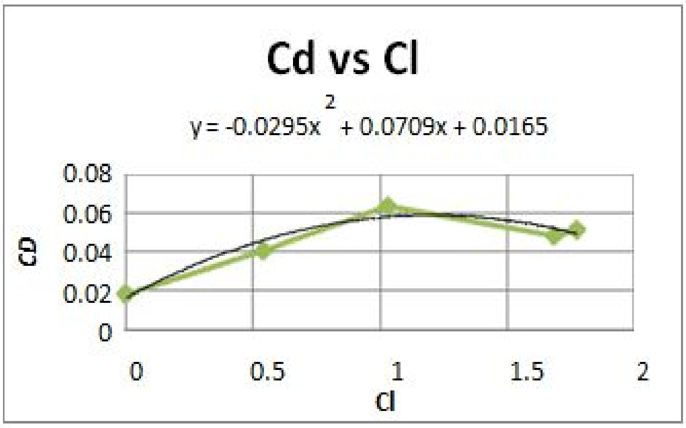

Figure 18 Cd vs Cl

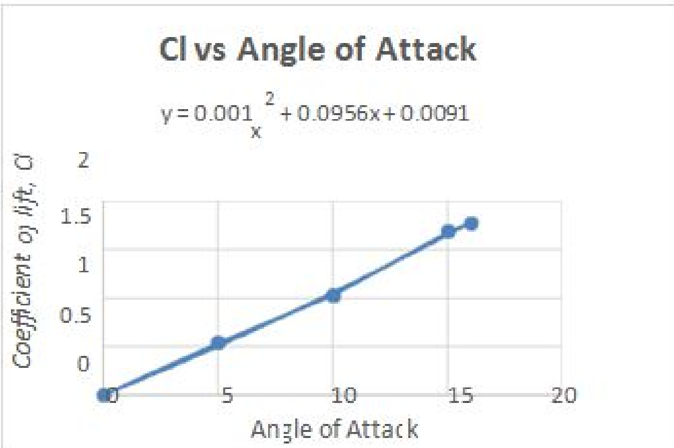

Figure 19 Cl vs angle of attack

Table 1 Comparison of Theoretical and calculated values of properties [1][5]

Theoretical			Calculated 1,35,000 cell		Error %	
α	Cl	Cd	Cl	Cd	Cl	Cd
0	0	0.0156	0.0002	0.01818	-	16.55769
5	0.5483	-	0.53917	0.04063	-1.66514	-
10	1.0966	-	1.0286	0.06364	-6.20098	-
15	1.6449	-	1.6835	0.04845	2.346647	-
16	-	-	1.7714	0.05145	-	-

4. CONCLUSIONS

In this lift and drag performances of NACA 0015 airfoil were performed. A ANSYS FLUENT program was used to numerical calculations. Numerical and experimental results were compared. The inference from calculated results is as follows:

- Drag and lift coefficients increased with increasing angle of attack.
- Stall was started with 16° attack angle. Lift coefficient increased whereas; drag coefficient increased.
- The optimum lift coefficient value was computed at16°

REFERENCES

[1] An essay by Steven D. Miller, on "Lift, Drag and Moment of a NACA 0015 Aerofoil", published on 28 May,2008.

[2] Karrna .S. Patel, Saumel B. Patel, Utsav B. Patel, Prof. Ankit P. Ahuja, UVPCE, Ganpat University, CFD Analysis of Airfoil, International journal of engineering research, volume 3,Issue No.3, page: 154-158.

[3] Developing Wind Power Projects: Theory and Practice By ToreWizelius.

[4] http://airfoiltools.com/airfoil/naca4digit..

[5] Haritonidis, J. H., "Lift, Drag and Moment of a NACA 0015 Aerofoil". Aerospace Engineering, The Ohio State University, Columbus,2008.

[6] İzzet Şahin and Adem Acir, 2015, "Numerical and Experimental Investigations of Lift and Drag Performances of NACA 0015 Wind Turbine aerofoil". International Journal of Materials, Mechanics and Manufacturing, Vol. 3, No.1, pp 22- 25.

[7] Kevadiya ,M.," CFD Analysis of Pressure Coefficient for NACA 4412", International Journal of Engineering Trends and Technology,Vol. 4, No. 5, 2013, pp.2041-2043.

[8] N. Bhagat and Shashi Kant, Amit Tiwari, Advanced Tool for Fluid Dynamics-CFD and its applications in Automotive, Aerodynamics and Machine Industry. International Journal of Mechanical Engineering and Technology, 7(2), 2016, pp. 177–186.

[9] Shekhar Dinkar Thakre, Jayashree P. Zope, Nilima A. Bachchuwar and Sourabh S. Kulkarni, Analysis of Straight Microchannel Heat Sink Using Computational Fluid Dynamics. *International Journal of Mechanical Engineering and Technology,* 7(4), 2016, pp. 234–242.